
BiaxialPRO – Complex cross-section analysis

What's new?

Version 3.3

STRUANG
Studio Tecnico Ing. Alessandro Angelini

Via Unterer Wiesenweg 36
Prissian - Tesimo
39010 (BZ)
ITALY

E-mail: alessandro@struang.com
Web: www.struang.com

Contents

Contents	2
General	3
Disclaimer of Warranty	3
Version	3
Description	4
BiaxialPRO main features	4
General characteristics	5
Cross-sections types	5
Main characteristics	5
Publications	6
Previous version	6
New Version	6
What's new?	7
List of New Features and Improvements	7
Enhanced Algorithms for Reading CAD DXF Drawings	7
Powerful Iterative Calculation Routines for Composite Columns	8
New Materials According to the 2020 Eurocodes	9
New Functionality for Profile Strength Assessment Based on Thickness	9
“Dialog-Free” Mode for Faster Calculation Execution	10
Example Sections Folder Accessible from the Top Bar	10
Interface Optimization with a Dynamic Tree Menu	10
Conclusion	10
Novità della versione?	11
Elenco delle nuove implementazioni	11
Potenziamento degli algoritmi di lettura dei disegni CAD in formato DXF	11
Implementazione di nuove routine di calcolo iterativo per sezioni miste di colonne	12
Introduzione di nuovi materiali aggiornati agli Eurocodici 2020	13
Nuova funzionalità per la valutazione della resistenza del profilato in funzione dello spessore	13
Modalità di calcolo senza finestre di dialogo	14
Inserimento della cartella esempi nella barra superiore	14
Ottimizzazione dell'interfaccia con menu ad albero dinamico	14
Conclusione	14

General

Disclaimer of Warranty

Struang - Studio Tecnico Ing. Alessandro Angelini (Author) makes no warranties or representations with respect to the contents or performance of the BiaxialPRO package or this manual. It particularly disclaims any implied warranty of fitness for any particular purpose.

The BiaxialPRO package is provided "as is" with all faults. Any claims made by sales literature or salespersons do not constitute warranties. Because of the diversity of hardware, software and conditions under which the system may be used, Author cannot make any warranty of fitness for a particular purpose. The entire risk of using the software must be assumed by the user. Accordingly, the user is recommended to thoroughly test the software before relying on it. In any event, any liability of Author is excluded.

It is the user's responsibility to ensure that they conform to any laws concerning the changes and restrictions they are permitted to make to an end user's computer and its software.

Struang - Studio Tecnico Ing. Alessandro Angelini reserves the rights to revise and to make changes to the software and and/or this manual without incurring any obligation to notify any person of such changes and revisions. There are no warranties, explicit or implicit, that the codes of the software are free of minor error, or are consistent with any particular standard of accuracy, or that they will meet user requirements for any particular application. They should not be relied on for any purpose where incorrect results could result in loss of property or personal injury. The use of BiaxialPRO software for any such purpose it is at user risk. The author, Ing. Alessandro Angelini, disclaim all liability of any kind, direct or consequential, resulting from usage/use by the user of this code. If user find that something does not seems to work correctly with the BiaxialPRO software, he could email a notification of problem to struang.engineering@gmail.com. If user use BiaxialPRO to produce scientific articles, we would appreciate being informed of the article's title, authors and place of appearance.

2025 Struang - Studio Tecnico Ing. Alessandro Angelini this publication and BiaxialPRO package:

All rights are reserved.

No part of this publication or the BiaxialPRO computer program to which it refers may be reproduced, transcribed, transmitted, stored in a retrieval system, or translated into any language, in any form or by any means mechanical, manual, electronic, magnetic, chemical, optical, or otherwise, except so far as described in this manual or accompanying Licence Agreement, without the prior written permission of Struang - Studio Tecnico Ing. Alessandro Angelini. Successful prosecution for unauthorized reproduction can result in imprisonment and/or fines.

BiaxialPRO package and its manual belonging to Struang - Studio Tecnico Ing. Alessandro Angelini.

Version

This document refers to BiaxialPRO version 3.3

Manual printed Oct 2025 for BiaxialPRO version 3.3

Description

BiaxialPRO software provides serviceability and ultimate analysis of a reinforced and composite cross-sections with standard and complex geometry.

BiaxialPRO main features

- Analysis
 - Linear and non-linear analysis of a generic non-homogeneous cross-section (concrete-steel-timber-carbon-glass fibers)
 - Staged construction analysis for composite cross-section (I.e. composite deck beams)
 - Columns 2nd order effects calculations
 - Beam analysis tool (Beam and cantilever scheme for any point force and distributed constant and variable linear loads, up to 6 load cases combinations)
- Geometry
 - Cross-sections geometry: standard rectangular, circular, “T”, non-standard hollow sections, non-symmetrical composite sections, hollow infilled composite sections with embedded profiles, several complex geometries defined on CAD environment.
 - Geometrical calculation of cross-section properties (i.e. section centroid and ellipse of inertia)
 - Cross-section wizard for standard RC and composite cross-sections
- Materials
 - Structural Steel material according EN1993-1-14: elastic-plastic and elastic-plastic with linear hardening
 - Concrete material according EN1992-1-1 and EN1992-1-1: parabola-rectangle stress-strain law include option for modified parabola-rectangle in compression for Annex L EN1992-1-1
 - Rebar according EN1992-1-1 and EN1992-1-1 : elastic-plastic and elastic-plastic with linear hardening
 - Different materials for rebars
 - Concrete confinement according to UNI-EN 1992-1-1
- Output
 - Stress and strain coloured map for any type of analysis (matrix material reacting or not to the tensile stress), with the capability to show a section cut for a specific plan of the cross-section
 - Ultimate cross-section resistance 3D domain with all the external forces plotted against the domain. This allows a quick verification of all the external forces in terms of ultimate resistance
 - Ultimate cross-section resistance 2D domain for a specified set of forces (M_x, M_y and N)
 - Moment vs Curvature diagram for any Biaxial bending direction. This graph shows the resistance of the cross section by increasing the curvature, showing all the relevant achieved point during the element loading (I.e. concrete spalling, reinforced yielded, concrete plasticisation, failure)
 - Ductility calculation for seismic request UNI EN 1998 (ITA annex)
 - Crack-width calculation (UNI EN 1992-1-1 - method1: based on rebate stress, diameter and distances inside the tensile zone; method2: rebar stress based)
 - Shear studs connection for a generic non-homogeneous composite cross-section (included timber/concrete)
 - Fatigue calculation output for composite deck cross-sections
 - Detailed report of all the relevant calculation (excel and word documents). Report Lang.: English, Italian and German. Report Format: Word, Excel and .txt
- Verifications
 - UNI EN 1992-1-1 SLS and ULS verification. A general verification panel shows the cross-section utilization factor for any verification type and for all the combinations (unlimited)
 - Minimum and maximum amount of reinforcement according EN1992-1-1

- Import/export
 - Import and export of un-limited forces combinations
 - Import and export of reinforcement
 - Import export of cross-section dxf (CAD)
 - Cross-section mesh export in dxf

General characteristics

BiaxialPro is a specialized software with a user interface designed by an engineer for engineers. The cross-section geometry can be defined using the interface, for standard simple sections, or using a Cad software (AutoCAD, draft sight etc..). For the latest case all the geometrical parameter are defined inside the drawing Cad environment using a specific set of layer for any element (rebars, edge, openings, embedded profiles). Therefore, BiaxialPRO generates a high defined mesh of the cross-section representing all the “micro fiber”. It associates to each micro fiber the property of the relative material. The calculations then follows the structural theory using integration formulation. In general, for any Biaxial bending direction, it calculates the cross-section response/reaction direction

Calculation can be performed assuming the matrix, in general concrete or timber or whatever confined a tensile fiber (i.e., gfrp etc.), reacting to the tensile stress or not reacting (I.e. cracked concrete)

Cross-sections types

- Reinforced concrete
- Composite
- Un-reinforced*

For the composite cross-sections it allows full embedded, partial embedded or external steel profiles.

* (with a specific definition)

Main characteristics

BiaxialPRO provides several graphs and diagrams output using Matlab libraries.

Output information consists of:

- Figure of stresses, strain diagrams
- Section geometry
- Section mesh (picture and dxf file)
- Centroid and ellipse of inertia
- Diagrams
- Stress strain relations for rebar and matrix material
- 3D interaction domain (N Mx My)
- 2D interaction domain
- Moment Vs curvature diagram
- Stress diagram using colormap for each analysis type and each combination
- Calculation Report
- All mechanical, geometrical, forces and settings assumptions
- Geometry information (i.e. reinforcement position, Inertia, neutral axis, etc)
- Crack width calculation details based on 7.3.4 EN 1992-1-1
- Maximum and minimum stress for each combination
- Fatigue calculations detail
- Shear connectors calculation details
- 3D stress distribution
- Curvature details with failure tables (yielding and plastic achievement at any curvature) (release 2.0)

Publications

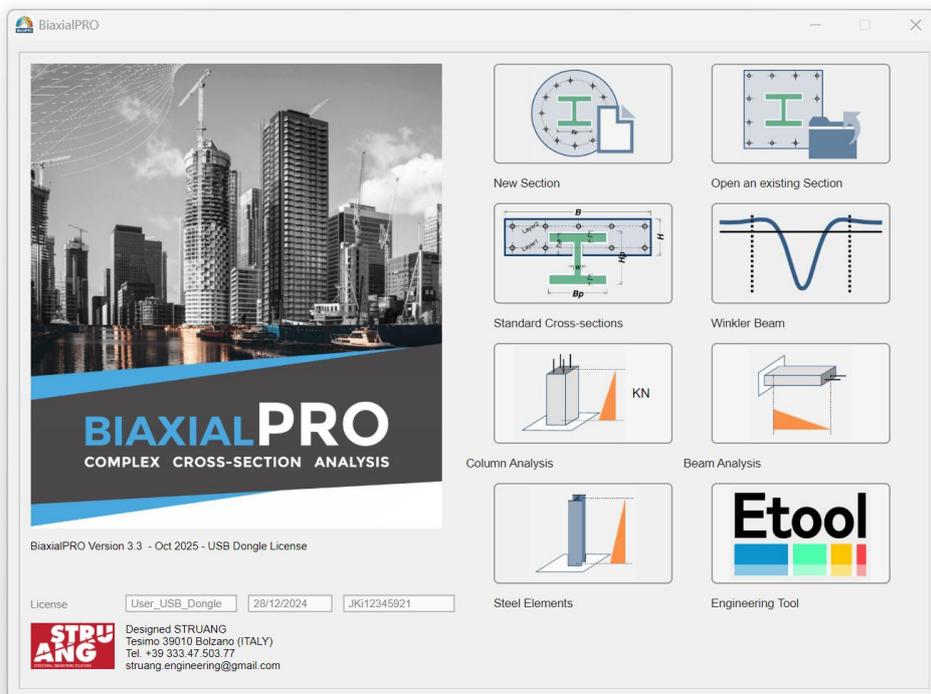
- BiaxialPRO has been used for the analysis of rc cores and steel-concrete composite cores presented in the technical paper called "Costruzioni Metalliche" of January 2021
- BiaxialPRO has been presenting during the latest Italian steel conference in participation with the partner Arcelor Mittal Luxembourg

Previous version

BiaxialPRO Software - Version 3.2 released Jun 2025.

New Version

BiaxialPRO Software - Version 3.3 released Oct 2025.



What's new?

BiaxialPRO software has been enhanced with several implementations and improvements included in the new version called 3.3. The main changes are listed in the following paragraphs.

List of New Features and Improvements

1. Enhanced algorithms for reading CAD DXF drawings
2. Implementation of powerful iterative calculation routines for composite column sections
3. Addition of new materials compliant with the 2020 Eurocodes
4. New functionality to assess profile strength as a function of thickness
5. New “dialog-free” mode to speed up calculations
6. A new section in the top bar provides access to ready-to-use DXF section examples
7. Interface optimization with a dynamic tree menu replacing side panel buttons

Enhanced Algorithms for Reading CAD DXF Drawings

The latest version of the software introduces significant improvements in the way CAD drawings in DXF format are interpreted. The parsing algorithms have been reinforced, allowing the software to correctly interpret sections even when they are not perfectly drawn—for instance, when two lines nearly meet but do not exactly intersect. Users can now define a node merging tolerance, allowing points within a certain proximity to be automatically merged into a single node.

Additionally, the software can now automatically correct drawing errors, such as overlapping lines or minor discontinuities. This eliminates the previous constraint requiring users to draw each boundary in one continuous motion, without lifting the mouse. Despite these enhancements, naming conventions for layers remain unchanged, and only lines and points are accepted as valid drawing entities—no other geometries are supported.

Another valuable improvement is the ability to import sections generated from exploded polylines, a functionality that was not available in previous versions, increasing flexibility in preparing geometry.

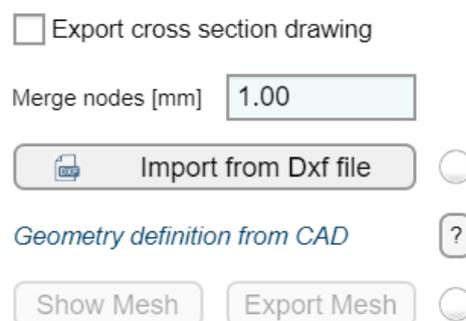


Figure 1 - Custom value for the merge distance between nodes

Powerful Iterative Calculation Routines for Composite Columns

This version integrates a highly efficient set of iterative calculation routines, specifically designed for analyzing composite column sections. These routines enable users to automatically run a wide range of parametric studies, adjusting variables such as section geometry, embedded steel profiles, reinforcement quantities, and material properties (concrete, reinforcing bars, and steel profiles).

The system can launch multiple calculation processes in parallel, significantly reducing computation time and automatically saving the output files with unique names. This functionality is particularly useful for users who need to compare design alternatives or carry out extensive optimization studies.

```

ROUTINE ANALYSIS INPUT DATA FILE GUIDE - COMPOSITE COLUMNS - METHOD B
V3.1 - 2025.09.24

-----
BEFORE LAUNCH THE ROUTINE
-----
This routine is based on several assumptions including the concrete materials
defined as per EN 1992-1-1:2003 and two cases of reinforcement percentage (1% and
4%).
Before user launch the iteration process, to save calculation time, please:
1.1 Adjust the mesh size = 20 or more (depending by the calculation speed/results
accuracy ratio you would like to reach.
1.2 In the "General" panel, option called "Interval angle of 3D resistance
domain" change the default value from 15° to 45°.

-----
general_info.txt
-----
Row 1 = List of steel profile Section Name (text)
Row 2 = List of Steel Grades of Embedded profiles
Row 3 = List of Steel Grades of Reinforcement material
Row 4 = List of Concrete Grades
Row 5 = List of Reinforcement percentages (text). i.e. 1% 4%

-----
profiles_data.txt
-----
The matrix components represents:
Row 1 = Steel profile Height
Row 2 = Steel profile width
Row 3 = Flange thickness
Row 4 = Web thickness
Columns 1 = First profile ... Columns "n" = n-th profile

-----
rebars_num_1.txt (p=1%)
-----
The matrix components represents:
Row 1 = Rebars Diameter
Row 2 = Number of rebars along side 1 - y direction
Row 3 = Number of rebars along side 2 - y direction
Row 4 = Number of rebars along side 3 - z direction
Row 5 = Number of rebars along side 4 - z direction
Columns 1 = First profile ... Columns "n" = n-th profile

-----
rebars_num_4.txt (p=4%)
-----
Same as previous file "rebars_num_1.txt"

-----
concrete_materials.txt
-----
The txt file contains all the parameters applied to the CONCRETE materials
according
Each row represents the value of a specific parameter, for both design and mean
values, as defined below:
Concrete material and code parameters - DESIGN VALUES
Row 1 = Ecm Concrete Elastic Modulo [Npa]
Row 2 = fck Concrete characteristic resistance as defined in EN1992-1-1:2003 [-]
Row 3 =  $\gamma_c$  is the reduction factor of the concrete characteristic resistance [-]
Row 4 =  $\gamma_{pr}$  is the reduction factor of the concrete characteristic resistance [-]
Row 5 = fck Cubic characteristic resistance of concrete [MPa]
Row 6 = fck Cylindrical characteristic resistance of concrete [MPa]
Row 7 = fcd Cylindrical design resistance of concrete [MPa]
Row 8 = fcm Cylindrical mean resistance of concrete [MPa]
Row 9 =  $\epsilon_{cu}$  concrete strain limit as per EN 1992-1-1 [-]
Row 10 =  $\epsilon_{cu2}$  concrete strain ultimate limit as per EN 1992-1-1 [-]
Concrete material and code parameters - MEAN VALUES
Row 11 = Ecm [Npa]
Row 12 =  $\gamma_c$  [-]
Row 13 =  $\gamma_{pr}$  [-]
Row 14 =  $\gamma_{pr}$  [-]
Row 15 = fck [MPa]
Row 16 = fck [MPa]
Row 17 = fcd [MPa]
Row 18 = fcm [MPa]
Row 19 =  $\epsilon_{cu}$  [-]
Row 20 =  $\epsilon_{cu2}$  [-]
Columns 1 = First Concrete Property ... Columns "n" = n-th Concrete Property

-----
rebar_materials.txt
-----
The txt file contains all the parameters applied to the REBARS materials according
Each row represents the value of a specific parameter, for both design and mean
values, as defined below:
Rebar material and code parameters - DESIGN VALUES
Row 1 = Esr Elastic Modulo of Reinforcement material [Mpa]
Row 2 =  $\gamma_s$  is the reduction factor of rebar, characteristic resistance [-]
Row 3 = fyk Characteristic yield strength of reinforcement [MPa]
Row 4 = fyk Design yield strength of reinforcement [MPa]
Row 5 =  $\rho_s$  is the density of rebar material for self-weight calculation [KN/m³]
Row 6 =  $\epsilon_{sd}$  elastic strain of reinforcement material [-]
Row 7 =  $\epsilon_{sd}$  ultimate strain of reinforcement material [-]
Row 8 = fud Design ultimate strength of reinforcement [MPa]
Rebar material and code parameters - MEAN VALUES
Row 9 = Esr [Mpa]
Row 10 =  $\gamma_s$  [-]
Row 11 = fyk [MPa]
Row 12 = fyk [MPa]
Row 13 =  $\rho_s$  [KN/m³]
Row 14 =  $\epsilon_{sd}$  [-]
Row 15 =  $\epsilon_{sd}$  [-]
Row 16 = fud [MPa]
Columns 1 = First Rebar Property ... Columns "n" = n-th Rebar Property

-----
cover_to_rebars.txt
-----
The matrix components represents:
Row 1 = c1 Distance from concrete edge to rebar axis along side 1
Row 2 = c2 Distance from concrete edge to rebar axis along side 2
Row 3 = c3 Distance from concrete edge to rebar axis along side 3
Row 4 = c4 Distance from concrete edge to rebar axis along side 1
Columns 1 = First profile ... Columns "n" = n-th profile

-----
cover_to_steel.txt
-----
The matrix components represents:
Row 1 = Cover to steel embedded profile cz (vertical direction)
Row 2 = Cover to steel embedded profile cy (horizontal direction)
Columns 1 = First profile ... Columns "n" = n-th profile
    
```

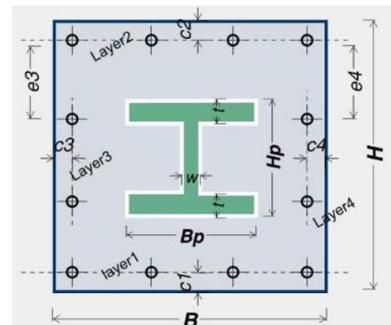


Figure 1 - Composite column section with definitions

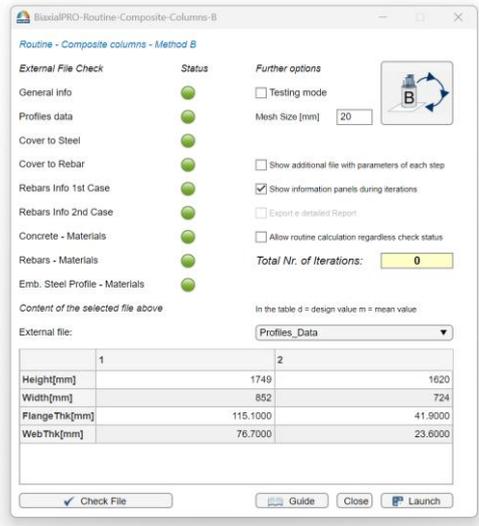


Figure 2 - Iteration process for composite columns. Main panel.

New Materials According to the 2020 Eurocodes

The new version includes an updated material library that reflects the latest Eurocode standards (2020 edition). New materials cover a broad spectrum of concrete grades, reinforcing bars, and steel profiles, allowing for more accurate modelling and compliance with current regulations. This ensures that both design and verification processes align with the latest engineering standards.

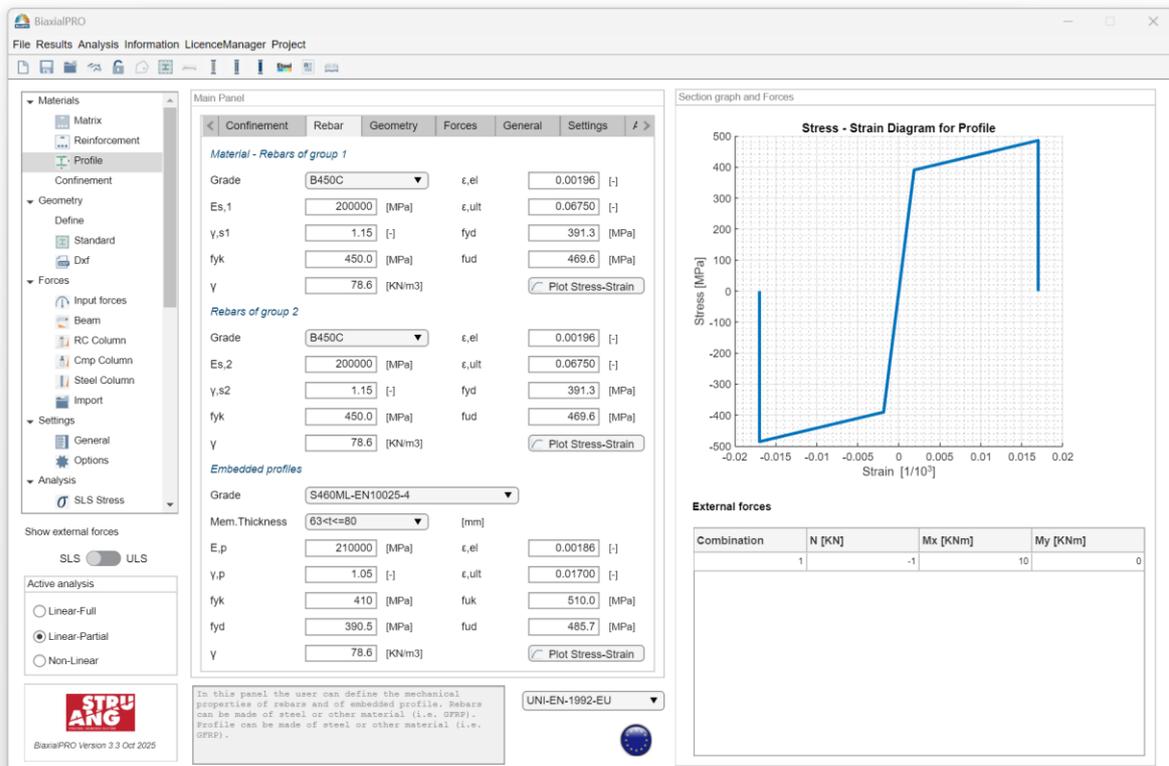


Figure 3 - Material definitions with additional materials and feature

New Functionality for Profile Strength Assessment Based on Thickness

A new analytical tool has been introduced, allowing users to evaluate the strength of steel profiles based on varying thicknesses. The software performs automatic, iterative analysis to determine how structural resistance changes as the profile thickness changes. This tool is especially valuable during early design stages or when selecting standard profiles, as it enables strength-to-weight and cost optimization.

“Dialog-Free” Mode for Faster Calculation Execution

To further streamline workflows, a new “dialog-free” calculation mode has been added. When enabled, the software runs without interrupting the process with dialog boxes or user prompts, making it ideal for batch processing and automated iterative analysis. This enhancement significantly boosts productivity, especially in high-volume simulation environments.

Example Sections Folder Accessible from the Top Bar

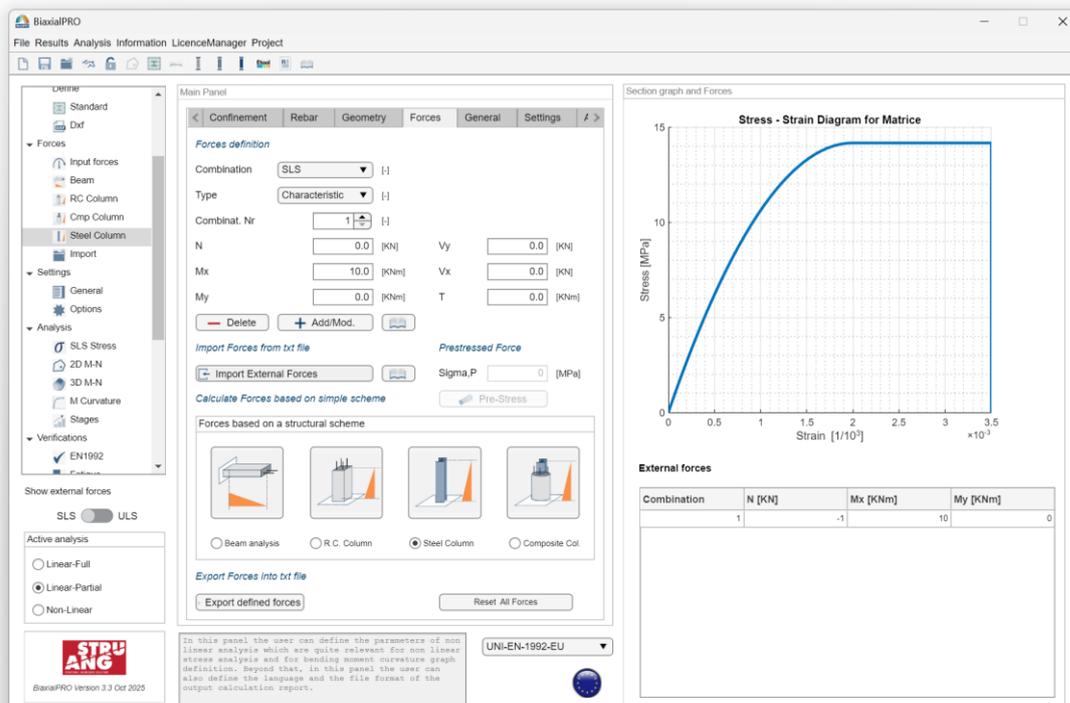
A new menu item in the top toolbar now provides direct access to a folder of pre-built DXF section examples. These examples serve as practical references to help users understand how to correctly build new sections. Especially useful for new users, this feature provides ready-to-use templates and visual guides, reducing errors and accelerating the learning curve.

Interface Optimization with a Dynamic Tree Menu

The user interface has been thoroughly redesigned for improved navigation and usability. The old left-hand panel with fixed buttons has been removed and replaced with a dynamic tree menu, offering quick access to different commands and environments within the software. This allows users to launch operations more efficiently and move fluidly across modules, resulting in a more intuitive and productive experience.

Conclusion

These improvements mark a substantial upgrade in both functionality and user experience. By combining advanced automation, increased CAD flexibility, Eurocode compliance, and interface enhancements, the new version of the software is more powerful, efficient, and user-friendly than ever before, designed to meet the evolving needs of engineers and designers.



Novità della versione?

BiaxialPRO software è stato migliorato con molte implementazioni e miglioramenti inclusi nell'ultima versione denominata 3.3. Le principali implementazioni vengono prima elencate brevemente e poi mostrate nel dettaglio.

Elenco delle nuove implementazioni

- Potenziamento degli algoritmi di lettura dei disegni CAD in formato DXF
- Implementazione di nuove routine di calcolo iterativo per sezioni miste di colonne
- Introduzione di nuovi materiali aggiornati agli Eurocodici 2020
- Nuova funzionalità per la valutazione della resistenza del profilato in funzione dello spessore
- Modalità di calcolo senza finestre di dialogo per ottimizzare i tempi di elaborazione
- Inserimento di una cartella esempi direttamente accessibile dalla barra superiore
- Ottimizzazione dell'interfaccia con l'introduzione di un menu ad albero dinamico

Potenziamento degli algoritmi di lettura dei disegni CAD in formato DXF

L'ultima versione del software ha visto un notevole miglioramento nella gestione e lettura dei file CAD in formato DXF. Gli algoritmi di parsing sono stati resi molto più robusti e ora riescono a interpretare correttamente sezioni disegnate anche in modo imperfetto. Una delle innovazioni principali è la possibilità per l'utente di definire una tolleranza di unione dei nodi ("nodes merging"), grazie alla quale elementi che non coincidono perfettamente – come due linee che si avvicinano ma non si toccano esattamente – possono essere riconosciuti come connessi.

Inoltre, il software è ora in grado di correggere automaticamente errori di disegno, come sovrapposizioni o intersezioni incoerenti tra linee. Questa nuova intelligenza evita la necessità di disegnare con estrema precisione e continuità, superando la precedente esigenza di "non staccare la mano dal mouse" durante la creazione di un bordo. Rimangono tuttavia invariate le convenzioni relative ai nomi dei layer, così come la restrizione all'uso esclusivo di linee e punti all'interno del disegno.

Un altro aspetto molto rilevante è la possibilità di importare sezioni ottenute da polilinee esplose, una funzionalità precedentemente non supportata, che aumenta notevolmente la flessibilità nella preparazione dei disegni.

Export cross section drawing

Merge nodes [mm]

Geometry definition from CAD

Implementazione di nuove routine di calcolo iterativo per sezioni miste di colonne

È stata integrata una potente suite di routine di calcolo iterativo, pensata per affrontare con efficacia l'analisi di sezioni miste di colonne. Le nuove routine permettono di automatizzare una vasta gamma di analisi parametriche, modificando rapidamente configurazioni relative a geometria della sezione, tipologia e dimensioni del profilato annegato, quantità di armatura, nonché le proprietà meccaniche dei materiali impiegati (calcestruzzo, acciaio delle barre e dei profilati).

Grazie a una gestione efficiente dei processi, il software può eseguire numerose simulazioni in parallelo, con salvataggio automatico dei risultati in file di output rinominati in modo ordinato. Questo rappresenta un enorme passo avanti per chi desidera effettuare analisi comparative o ottimizzazioni strutturali in tempi contenuti.

ROUTINE ANALYSIS INPUT DATA FILE GUIDE - COMPOSITE COLUMNS - METHOD B
V3.1 - 2025.09.24

BEFORE LAUNCH THE ROUTINE

This routine is based on several assumptions including the concrete materials defined as per EN 1992-1-1:2003 and two cases of reinforcement percentage (1% and 4%).
Before user launch the iteration process, to save calculation time, please:
1.1 Adjust the mesh size = 20 or more (depending by the calculation speed/results accuracy ratio you would like to reach).
1.2 In the "General" panel, option called "Interval angle of 3D resistance domain" change the default value from 15° to 45°.

general_info.txt

Row 1 = List of steel profile Section Name (text)
Row 2 = List of Steel Grades of Embedded profiles
Row 3 = List of Steel Grades of Reinforcement material
Row 4 = List of Concrete Grades
Row 5 = List of Reinforcement percentages (text). i.e. 1% 4%

profiles_data.txt

The matrix components represents:
Row 1 = Steel profile Height
Row 2 = Steel profile width
Row 3 = Flange thickness
Row 4 = Web thickness
Columns 1 = First profile ... Columns "n" = n-th profile

rebars_num_1.txt (p=1%)

The matrix components represents:
Row 1 = Rebars Diameter
Row 2 = Number of rebars along side 1 - y direction
Row 3 = Number of rebars along side 2 - y direction
Row 4 = Number of rebars along side 3 - z direction
Row 5 = Number of rebars along side 4 - z direction
Columns 1 = First profile ... Columns "n" = n-th profile

rebars_num_4.txt (p=4%)

Same as previous file "rebars_num_1.txt"

concrete_materials.txt

The txt file contains all the parameters applied to the CONCRETE materials according
Each row represents the value of a specific parameter, for both design and mean values, as defined below:

Concrete material and code parameters - DESIGN VALUES
Row 1 = Ecm Concrete Elastic Modulo [Npa]
Row 2 = fck Concrete characteristic strength of concrete [MPa]
Row 3 = α_{cc} is the reduction factor of the concrete characteristic resistance [-]
Row 4 = γ_c is the reduction factor of the concrete characteristic resistance [-]
Row 5 = fctk Concrete characteristic tensile strength of concrete [MPa]
Row 6 = fctk Cylindrical characteristic resistance of concrete [MPa]
Row 7 = fcd Cylindrical design resistance of concrete [MPa]
Row 8 = fcm Cylindrical mean resistance of concrete [MPa]
Row 9 = ϵ_{ck} concrete strain limit as per EN 1992-1-1 [-]
Row 10 = ϵ_{cm} concrete strain ultimate limit as per EN 1992-1-1 [-]

Concrete material and code parameters - MEAN VALUES

Row 11 = Ecm [Npa]
Row 12 = fcm [-]
Row 13 = α_{cc} [-]
Row 14 = γ_c [-]
Row 15 = fctk [MPa]
Row 16 = fctk [MPa]
Row 17 = fcd [MPa]
Row 18 = fcm [MPa]
Row 19 = ϵ_{ck} [-]
Row 20 = ϵ_{cm} [-]

Columns 1 = First Concrete Property ... Columns "n" = n-th Concrete Property

rebar_materials.txt

The txt file contains all the parameters applied to the REBARS materials according
Each row represents the value of a specific parameter, for both design and mean values, as defined below:

Rebar material and code parameters - DESIGN VALUES
Row 1 = Esr Elastic Modulo of Reinforcement material [Mpa]
Row 2 = γ_s is the reduction factor of rebar, characteristic resistance [-]
Row 3 = fyk Characteristic yield strength of reinforcement [MPa]
Row 4 = fyd Design yield strength of reinforcement [MPa]
Row 5 = ρ_s is the density of rebar material for self-weight calculation [KN/m³]
Row 6 = ϵ_{sk} elastic strain of reinforcement material [-]
Row 7 = ϵ_{sk} ultimate strain of reinforcement material [-]
Row 8 = fud Design ultimate strength of reinforcement [MPa]

Rebar material and code parameters - MEAN VALUES

Row 9 = Esr [Mpa]
Row 10 = γ_s [-]
Row 11 = fyk [MPa]
Row 12 = fyk [MPa]
Row 13 = ρ_s [KN/m³]
Row 14 = ϵ_{sk} [-]
Row 15 = ϵ_{sk} [-]
Row 16 = fud [MPa]

Columns 1 = First Rebar Property ... Columns "n" = n-th Rebar Property

cover_to_rebars.txt

The matrix components represents:

Row 1 = c1 Distance from concrete edge to rebar axis along side 1
Row 2 = c2 Distance from concrete edge to rebar axis along side 2
Row 3 = c3 Distance from concrete edge to rebar axis along side 3
Row 4 = c4 Distance from concrete edge to rebar axis along side 1

Columns 1 = First profile ... Columns "n" = n-th profile

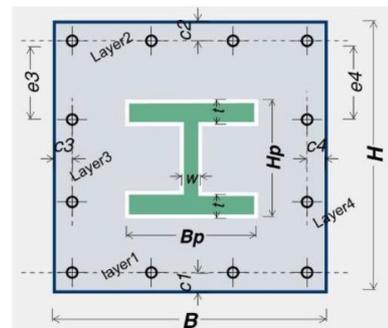


Figure 1 - Composite column section with definitions

cover_to_steel.txt

The matrix components represents:

Row 1 = Cover to steel embedded profile cz (vertical direction)
Row 2 = Cover to steel embedded profile cy (horizontal direction)

Columns 1 = First profile ... Columns "n" = n-th profile

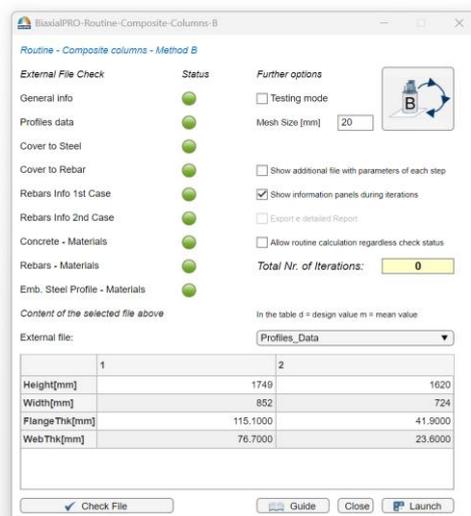


Figure 4 - Iteration process for composite columns. Main panel.

Introduzione di nuovi materiali aggiornati agli Eurocodici 2020

In ottemperanza agli aggiornamenti normativi, la nuova versione del software include una libreria di materiali aggiornata agli Eurocodici, con riferimento in particolare alle versioni più recenti del 2020. I materiali introdotti coprono un ampio spettro di resistenze e combinazioni utili sia per il calcestruzzo che per gli acciai da armatura e i profilati metallici, permettendo all'utente una modellazione più realistica e conforme alle normative vigenti.

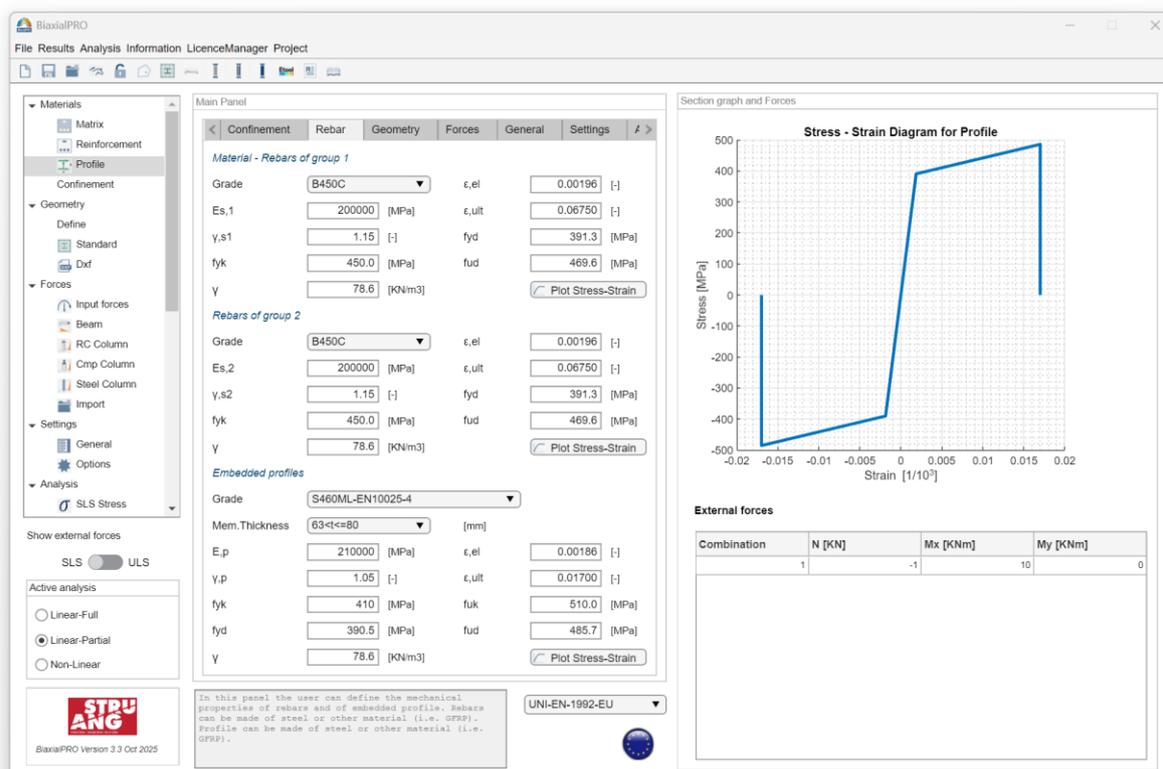


Figure 5 – Definizione di materiali aggiuntivi e nuova funzionalità per lo spessore

Nuova funzionalità per la valutazione della resistenza del profilato in funzione dello spessore

Una delle funzionalità più innovative riguarda la possibilità di valutare la resistenza del profilato variando lo spessore. Il software è ora in grado di calcolare, in maniera automatica e iterativa, come cambia la capacità resistente del profilato in funzione dello spessore adottato, consentendo una ottimizzazione spinta delle sezioni metalliche sia in termini strutturali che economici. Questa caratteristica si rivela particolarmente utile in fase di progettazione preliminare o nella selezione di profilati standard ottimizzati.

Modalità di calcolo senza finestre di dialogo

Per velocizzare ulteriormente le operazioni, è stata introdotta una modalità di calcolo priva di finestre di dialogo. Questa nuova modalità permette all'utente di eseguire calcoli in maniera diretta e continua, senza interruzioni dovute alla comparsa di finestre o richieste di conferma. L'implementazione risulta particolarmente efficace quando si eseguono calcoli batch o iterativi, in cui l'interazione dell'utente è minima e l'efficienza massima.

Inserimento della cartella esempi nella barra superiore

È stato aggiunto un accesso diretto, tramite la barra superiore del software, a una cartella contenente esempi di sezioni DXF già realizzate. Questo strumento ha l'obiettivo di guidare l'utente nella creazione di nuove sezioni, fornendo casi pratici e template da cui partire. La presenza di esempi preconfezionati rappresenta un importante supporto didattico e operativo, soprattutto per i nuovi utenti o per chi si confronta per la prima volta con sezioni complesse.

Ottimizzazione dell'interfaccia con menu ad albero dinamico

Infine, l'interfaccia utente è stata completamente ripensata per offrire una navigazione più snella ed efficace. I pulsanti presenti nel pannello laterale sinistro sono stati rimossi e sostituiti da un menu ad albero dinamico, che consente di esplorare rapidamente tutte le sezioni del software. Grazie a questo nuovo sistema, l'utente può accedere direttamente a funzionalità avanzate da qualsiasi ambiente operativo, migliorando la produttività e riducendo i tempi di accesso ai comandi principali.

Conclusione

Le nuove migliorie introdotte rappresentano un **salto di qualità sostanziale** sia dal punto di vista della fruibilità che delle prestazioni tecniche del software. L'attenzione alle esigenze degli utenti, l'aderenza alle normative e l'introduzione di funzionalità intelligenti e automatizzate rendono questa nuova versione uno strumento ancora più completo, versatile ed efficiente.

The screenshot displays the BiaxialPRO software interface. On the left, a dynamic tree menu is visible under the 'Forces' section, with 'Steel Column' selected. The main panel is titled 'Main Panel' and contains the 'Forces definition' section. It includes fields for 'Combination' (SLS), 'Type' (Characteristic), and 'Combinat. Nr' (1). Below these are input fields for forces: N (0.0 [kN]), Mx (10.0 [kNm]), My (0.0 [kNm]), Vy (0.0 [kN]), Vx (0.0 [kN]), and T (0.0 [kNm]). There are buttons for 'Delete', 'Add/Mod.', and 'Import External Forces'. The 'Import External Forces' section includes a 'Sigma.P' field (0 [MPa]) and a 'Pre-Stress' checkbox. Below this, there are icons for 'Beam analysis', 'R.C. Column', 'Steel Column' (selected), and 'Composite Col'. At the bottom of the main panel, there are 'Export Forces into txt file' and 'Export defined forces' buttons, along with a 'Reset All Forces' button. On the right side of the interface, a graph titled 'Stress - Strain Diagram for Matrice' is shown. The y-axis is 'Stress [MPa]' ranging from 0 to 15, and the x-axis is 'Strain [1/10³]' ranging from 0 to 3.5. The graph shows a blue curve that rises steeply from the origin, reaches a peak stress of approximately 14 MPa at a strain of about 2.0, and then levels off. Below the graph is a table titled 'External forces' with the following data:

Combination	N [kN]	Mx [kNm]	My [kNm]
1		-1	10